SmaggIce User Manual
version 1.8

NASA Glenn Research Center
21000 Brookpark Rd.
Cleveland, OH 44135

May 4, 2005

Table of Contents

Introduction

What is SmaggIce?

SmaggIce (Surface Modeling And Grid Generation for Iced Airfoils) is one of NASA's aircraft icing research codes developed at the Glenn Research Center. It is a software toolkit used in the process of aerodynamic performance prediction of iced airfoils. Its primary purpose is to create structured grids for 2D iced airfoils in preparation for Computational Fluid Dynamics (CFD) analysis. It provides tools which support and complement the 2D grid-based CFD process, allowing you to:

- measure ice shape characteristics
- prepare the ice surface for gridding
- perform domain decomposition for single-element airfoils
- create and modify grids
- analyze grid quality
- output grids for subsequent input into flow solvers

Release Information

The SmaggIce software is still under development. Version 1.8 is an intermediate release, and further capabilities are planned for inclusion in later releases. The software is furnished as is, with no warranty of fitness for any particular use. You are not permitted to give the SmaggIce code to any other organizations and/or persons outside your company. We would be interested in knowing about any problems or errors that you experience with the codes, as well as any new features you would like to see included in future releases.

Contact Information

We would be interested in knowing about any problems or errors that you experience with the software, as well as any new features you would like to see included in the future. Questions and comments can be sent via e-mail to:

`smaggice@grc.nasa.gov`

Access and Installation

System Requirements

SmaggIce is intended to run on UNIX or MS-Windows platforms. The GUI was developed for the X Window System. OpenGL is used for the graphics drawing. It uses the GLX extensions to X to interface with the windowing system. A non-OpenGL executable is available for use on UNIX workstations that do not have OpenGL. A non-GLX executable is available for use on MS-Windows workstations whose X server does not support the GLX extensions. These non-OpenGL/non-GLX versions use the Mesa library (which is distributed as part of the package) rather than the OpenGL library, but they do not run as fast as the OpenGL/GLX executables.

Currently, SmaggIce is only distributed as a binary executable. If you need a version of SmaggIce for a platform that is not included in the distribution, please contact the SmaggIce team.

UNIX Systems

SmaggIce was designed and written to run on any UNIX platform. It has been built for the following systems:

- SGI Irix 6.5 (32 bit) using X and OpenGL
- SGI Irix 6.5 (32 bit) using X without OpenGL
- SUN SunOS 5.8 using X and OpenGL
- SUN SunOS 5.8 using X without OpenGL
- Intel GNU/Linux 2.6.10 using X and OpenGL
- Intel GNU/Linux 2.6.10 using X without OpenGL

To run a SmaggIce on a UNIX platform, you should have the following:

- Web browser software (such as Netscape or Mozilla) for viewing help files
- If you are using the SmaggIce package that uses OpenGL, you must have the OpenGL libraries installed on your system.
- The Linux version requires access to the `libpgc.so` shared library, part of the Linux portability package from The Portland Group. It can be downloaded from `http://www.pgroup.com/support/download_release.php`

MS-Windows Systems

SmaggIce has been ported to run on MS-Windows systems. It has been built for the following systems:

- MS-Windows 2000/XP using X-server with GLX extension
- MS-Windows 2000/XP using X-server without GLX extension

To install and run SmaggIce on a PC running Microsoft Windows, you should have the following:

- Pentium-class PC (300MHz or higher recommended)
- 32 megabytes of Random Access Memory (64 MB of RAM recommended)
- disk space (MB = megabytes): 65 MB before installation and 30 MB after the install is complete
- Super VGA monitor (or better)
- standard graphics card (or better)
- Windows 2000 or Windows XP installed as the operating system
- administrative privileges on the system
- Microsoft Internet Explorer, Mozilla Firefox, or Netscape web browser software for viewing help files

- third party X-server software already installed on your PC (available from various sources)
- If you are using the SmaggIce package with GLX extensions, you will need OpenGL extensions (GLX) installed with your X-server software. (You must have already installed an X-server.) SmaggIce has been tested with several X-servers, including Hummingbird Exceed, HOBLink, MI/X 4.0, Wina/XE, and X-ThinPro. Some of these support GLX extensions directly, but for others, if you want to use GLX extensions, then you must install additional libraries (for example, Hummingbird's Exceed3D). One way to determine if your X-server supports the GLX extensions is to run the `xdpyinfo` command on an X-client connected to your X-server. If the server supports the GLX extensions, the output of this command will list "GLX" under the heading "number of extensions.

How to get SmaggIce

This computer program is available to any U.S. organization upon request. It is furnished on the condition that it will be used only within and for the U.S. organization which requests the software, and that it will not be transmitted to other organizations.

The software may be requested through NASA Glenn Research Center's Software Repository:

```
https://technology.grc.nasa.gov/software/
```

When you request the software, you will have to specify which package you will be downloading:

- UNIX
- MS-Windows with GLX
- MS-Windows without GLX

All the UNIX versions come bundled in a single distribution package. You can install on any supported UNIX platform from that package. At run-time, you can choose whether to run with OpenGL or without OpenGL. The MS-Windows distribution comes in 2 separate packages: the GLX package containing the SmaggIce executable that uses GLX extensions, and the non-GLX package containing the SmaggIce executable without GLX extensions (using instead only standard X routines). If your X-server does not support GLX extensions, you should choose the non-GLX package. If you have any questions about which package to request, contact the SmaggIce team.

Distribution

The distribution contains:

- README file with a description of the contents and directions for installation and startup.
- Executable SmaggIce version 1.8 for each supported system.
- Auxiliary files and libraries needed to set up and run SmaggIce.
- A separate post-processing utility used to link CGNS grid and solution files.
- This user manual in PDF format.
- On-line help files in HTML format.
- Sample input geometry files.
- Sample script files.

Installation

UNIX Installation

1. Download the file `smaggice18.tar.gz` to a folder on your local hard drive (for example, `/tmp`). This compressed file contains the SmaggIce for UNIX distribution.

2. Create a directory to hold the SmaggIce files and change to that directory. For example:
   `> mkdir $HOME/smaggice1.8`
   `> cd $HOME/smaggice1.8`

3. Uncompress and extract the downloaded files to this directory. For example;
   `> gunzip -c /tmp/smaggice18.tar.gz | tar xovf -`

4. Edit the file `bin/runsmagg`. Detailed directions are in the file.

   You will have to set at least two environmental variables in that file:
   `SMG_DIR` - directory into which the SmaggIce distribution was copied
   `SMG_VENDOR` – the architecture on which you are running

   On some systems, you have the option of displaying the graphics using OpenGL or X only (without OpenGL). The default is to run the OpenGL version, if it is available. If you would rather run the X-only version, that option can be set here.

   To display on-line help, you may need to set the variable `WEBBROWSER` to point to the executable that starts up your web browser.

MS-Windows Installation

To install SmaggIce on Windows XP, you must log onto an account with administrative rights.

1. Download the file `SmaggIce_v1.8.zip` to a folder on your local hard drive (for example, `C:\TEMP\SmaggIce`). This compressed file contains the SmaggIce for Windows distribution.

2. Unzip the downloaded file via WinZip (or other compatible decompression program) into the download folder (`C:\TEMP\SmaggIce`).

3. Double click on the file `setup.exe` in that download folder (`C:\Temp\SmaggIce\setup.exe`).

4. The install window will appear. Follow the directions that appear on the screen.

Uninstalling SmaggIce from a Windows PC

If you need to uninstall SmaggIce, use the *Add/Remove Programs* control panel (**Start button→ Settings→Control Panel→Add/Remove Programs**). Highlight the SmaggIce entry and click the **Remove** button. NOTE: You should always do this uninstall before re-installing SmaggIce. Do NOT try to install another SmaggIce package (GLX or non-GLX) without first uninstalling the package that is currently installed.

Starting SmaggIce

UNIX Startup

Before starting SmaggIce, be sure you have modified the `runsmagg` script (which can be found in the `smaggice1.8/bin` directory), as described in step 4 of the Installation instructions.

To run SmaggIce, invoke the `runsmagg` script in the `smaggice1.8/bin` directory by specifying its full pathname. For example, if you have installed SmaggIce in the directory `$HOME/smaggice1.8`, type:

```
> $HOME/smaggice1.8/bin/runsmagg
```

Alternatively, you may wish to add the `smaggice1.8/bin` directory to your path, then simply type:

```
> runsmagg
```

The SmaggIce main window will be displayed.

MS-Windows Startup

Follow these steps to run SmaggIce:

- If you will be running the GLX version of SmaggIce, make sure that the GLX extensions are enabled in your X-server software (see your X-server documentation);
- Start your X-server BEFORE running SmaggIce (consult your X-server documentation);
- Start SmaggIce via any of these alternatives:
  - double-click on the desktop shortcut,
  - click on the **Start button→Programs→SmaggIce→Run SmaggIce**, or
  - locate the file `smaggrun.bat` in Windows Explorer and double-click on it.

The SmaggIce main window will be displayed. If this main window does not appear within a few seconds, then check to be sure that you started the X-server before starting SmaggIce. If the on-line help does not work, then edit the file `smaggrun.bat`, and set the variable `WEBBROWSER` to point to the location of the executable that runs your web browser. If you get errors related to missing fonts, it may be that your X-server does not include the fonts that SmaggIce requires. In this case, follow the directions in the file `smaggice1.8\smgfonts\README`.

Using SmaggIce

Capabilities

SmaggIce provides three types of interactive software tools: measuring ice shape characteristics, preparing ice shapes for grid generation, and grid generation and modification. The ice shape characterization tool provides the means to measure the physical characteristics of ice such as icing limit locations, horn height and angle, and distance from the leading edge to any prominent ice location. The ice shape preparation tools provide the means to examine input geometry data, correct or modify any deficiencies in them, add artificial ice if desired, and perform a systematic smoothing of ice to a level that will make the CFD process manageable. The grid generation and modification tools provide the means to perform domain decomposition, grid generation, grid quality analysis, and grid modification.

Data Input

Point data defining a clean and/or iced airfoil 2D surface may be read into SmaggIce. Points on the surface of the airfoil should start at the trailing edge, wrap around either clockwise or counter-clockwise, and end up at the trailing edge. Data may also define an ice shape only, which would include only the points along the ice surface and not the points on the airfoil surface.

Measuring Ice Shape Characteristics

Ice shapes can be measured interactively by selecting data points on the screen, recording the measurements in a table, and saving them to a file. The types of measurements that can be made include:

- point location (e.g., ice limits)
- distance between two points (e.g., ice horn height or width)
- arc length (i.e., the sum of the lengths of the line segments along a boundary between two points on that boundary)
- angle between two lines (e.g., horn angle)
- (X-Xle)/C (i.e., (x-xLeadingEdge)/chordLength)
- ice area

Location and distance may be normalized by the chord length of a clean airfoil. Ice area may be normalized by the area of the clean airfoil. The points used in making the measurements can be selected by various methods: the closest point to any object, the closest point to the currently-selected object, the closest point to the reference airfoil, or an arbitrary point in space.

Preparing the Ice Surface for Gridding

Identify twists

SmaggIce will examine airfoil and ice geometry when it is read in and after modifications are made to it, to search for and identify twists (tangles) in the surface. These twists will be visually highlighted on the screen unless you turn that option off.

Close trailing edge

In some airfoil geometry, the trailing edge is not closed; i.e., the points on the upper and lower surface at the trailing edge do not coincide. In this case, you will have to extend the trailing edge to "close" the element before creating a grid. A simple menu pick will do this for you.

Extend ice to clean airfoil

For some ice shapes, the points on the complete airfoil are not included in the geometry. Rather, only the points of the ice accretion at the leading edge of the airfoil are included. In this case, you will have to read in the clean airfoil on which the ice is based and extend the ice to include the clean airfoil before creating a grid. A simple menu pick will do this for you.

Reference airfoil

Setting a reference airfoil allows the chord length to be calculated and the leading edge point to be identified. These values are needed for several SmaggIce functions, such as normalizing ice measurements or extending ice. A clean airfoil is usually read in and set as the reference airfoil. However, if clean airfoil geometry is not available, you may set an iced airfoil as the reference airfoil, and then set the set the chord length and leading edge point manually.

Boundary modification

The types of modifications that can be made to surfaces are curve smoothing, re-discretization to change the number and/or distribution of points, and reshaping. Any subcurve of an element can be selected for processing. Systematic smoothing of the iced boundaries in a controlled manner is accomplished using a control point formulation. Users can control the level of smoothing by choosing the number of control points in constructing curves. Curve discretization provides a means of increasing/decreasing the number of points, distributing the points by curvature, and controlling the uniformity of their distribution. In addition, hyperbolic tangent stretching is provided. Direct reshaping of the curve (to perhaps correct obvious ice shape errors) is done by dragging control points associated with the curve. These control features of SmaggIce not only prepare the ice surface for the grid-based CFD, but they also allow users to correct any deficiencies (e.g., tangles, gaps, too many or too few points) in the input data.

Adding artificial (computer-generated) ice

Adding computer-generated ice shapes to the surface of a clean airfoil is used to prepare for studies of the effects of surface roughness as well as the effect of various ice shapes on the airfoil aerodynamic performance. Once clean airfoil data is read in, the user may interactively add different types of geometries to the clean airfoil. These geometric ice shapes include: forward-facing right triangle, backward-facing right triangle, generic triangle, rectangle, forward-facing quarter circle, backward-facing quarter circle, semi-circle, and trapezoid. Parameters are used to define such things as the location, size, and number of points in the shapes.

Move Element

With this feature, the user can modify elements by rotating and/or translating the geometry. This may be useful to prepare for parametric studies on multi-element configurations. For instance an aileron can be translated and rotated about a user-specified hinge point.

Domain Decomposition and Grid Generation

The user can perform domain decomposition for a single-element airfoil. The domain will be multi-block and includes a block containing the Navier-Stokes viscous sublayer, near field block(s), and an outer block that overlaps the near field. Figure 1 below shows some parts of the decomposed domain. All blocks in the near field are abutting one-to-one. One-to-one connectivities between grids in the viscous sublayer and near field are generated automatically during domain decomposition. The overlap connectivities between the outerblock and near field block(s) are not calculated unless and until the grids are saved to a CGNS file. Boundary conditions are also generated automatically during domain decomposition. Figure 2 shows the boundary conditions that are defined for the different types of boundaries in the flow domain.

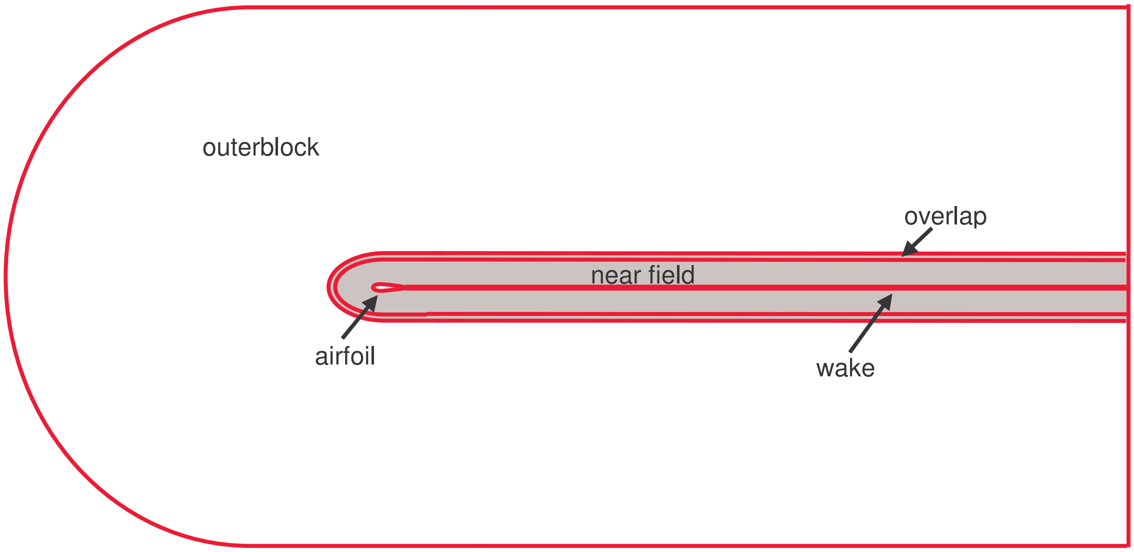


Figure 1. Domain decomposition divides the domain into multiple blocks which will be gridded. The near field block(s) are shown in gray and the outer block in white, but the viscous sublayer is too thin to be seen.

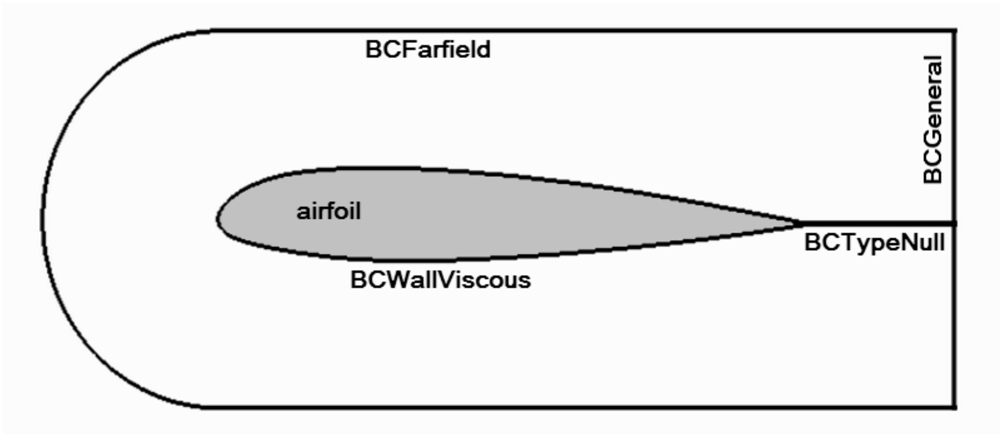


Figure 2. Boundary conditions defined at different types of boundaries: `BCFarfield` *along the outer boundary of the outerblock;* `BCWallViscous` *along the inner boundary of the viscous sublayer block where it surrounds the airfoil;* `BCGeneral` *on the exit boundary of the viscous sublayer, near field, and outer blocks, and* `BCTypeNull` *on all other boundaries.*

To perform the initial domain decomposition and grid generation, the user must:

- Identify the clean airfoil (or at least specify the chord length and the leading edge point).
- Select airfoil on which the grid will be based.
- Create a wake.
- Create a viscous sublayer block. This is a required, C-shape block, which wraps around the airfoil and extends back along the wake.
- Create the near field block(s). This includes at least a single “surrounding block”, but may include multiple blocks that surround the viscous sublayer. Figure 3 shows two types of near field decomposition: single block and multiple blocks with radial cuts.
- Create the outer block which overlaps the near field.

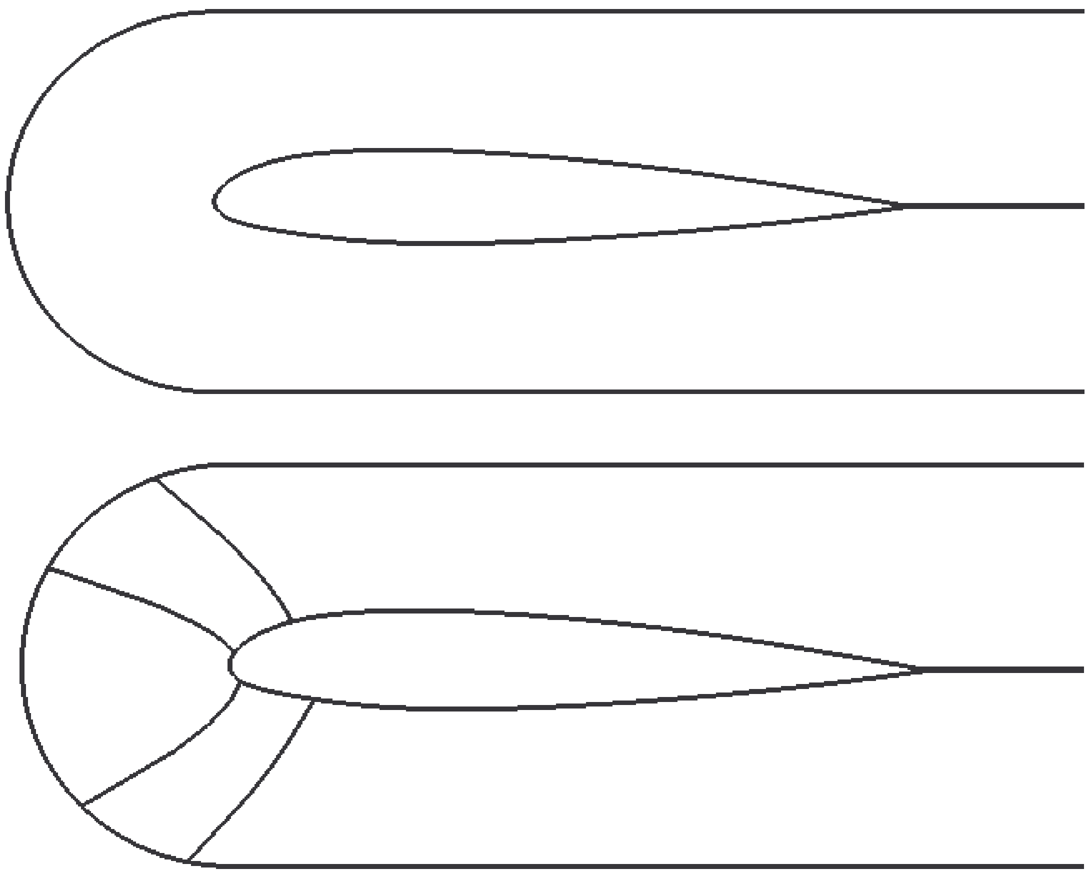


Figure 3. Top: single block near field; Bottom: multi-block near field created using radial cuts.

Modifying Block Boundaries and Grids

After the initial domain decomposition and grid generation, block boundaries may be modified to change the shape of the boundaries or change the point distribution. The number of points may not be changed at this time. This is to ensure that all blocks which are grid-ready and one-to-one abutting remain so even after modifications are made. Grids may be modified by:

- dividing along grid lines
- merging one-to-one abutting grids that share corners
- smoothing grids across shared edges of blocks
- performing interior smoothing
- setting cell spacing along an edge to match cell spacing of adjacent grids.

The user also has control of elliptic smoothing parameters used during grid generation. Changing these parameters will affect the appearance of the grid.

Analyzing Grid Quality

Grid quality measurements can be calculated and displayed graphically on the gridded blocks as color-coded overlays. This allows the user to quickly identify areas of poor grid quality, which may then be repaired. These grid quality measurements include:

- aspect ratio of grid cells

- orthogonality of grid cells
- skewness of grid cells
- stretching ratio in the I direction at each grid point
- stretching ratio in the J direction at each grid point
- grid cell shape; e.g., twisted, concave, or degenerate cells

Data Output

Grid Output

Once the grids are created, the user will want to output the grids for subsequent input into a flow solver. Two formats are available: PLOT3D and CGNS. PLOT3D files contain only the grid data. If PLOT3D files are used, another software package (such as GMAN) will be needed to define the boundary conditions and connectivity between the grids, since that information is not stored in a PLOT3D file. To avoid this extra step, CGNS files may be used for output. They already contain the connectivity information and boundary conditions as well as the grid data. CGNS files can be used as input directly into the Wind-US flow solver. SmaggIce also allows the user to create a simple Wind-US input data file which can be edited and used in conjunction with the CGNS file to control a Wind-US run.

Saving Images

The user can save the graphics display from SmaggIce as a GIF, TIFF, or PPM image file.

Save Extruded Wing

A 3-dimensional geometry file can be created by extruding iced airfoil geometry in the spanwise direction. Sinusoidal variations can be applied to the extrusion to create a true 3D geometry. This geometry cannot be used within SmaggIce, but it may be useful as input into other packages.

Save State

The current state of SmaggIce can be saved to a file. This allows the user to quit the application and upon restarting, restore the saved state. All settings and data are restored, so the user can continue working as if the session had not been interrupted.

Workflow

Figure 4 below illustrates the steps in the CFD process and the role of SmaggIce in that process. The SmaggIce role starts with input of an iced airfoil and ends with output of the grids, which are sent to a flow solver. The specific flow solver illustrated here is Wind-US. Further information on Wind-US is available from:

`http://www.grc.nasa.gov/WWW/winddocs/`

There are two paths for grid output. The preferred path is to save the grids to a CGNS grid file, since it also stores the boundary conditions and connectivities between the grid blocks. This can be used as direct input into Wind-US. An alternate path is to save the grids to a PLOT3D grid file. However, the boundary conditions and connectivities would have to be defined manually using a Wind-US pre-processor such as GMAN. SmaggIce can also create a Wind-US input file, which can be used as a basis for running a Wind-US job.

Many, if not all, post-processors including Fieldview, Ensight, and Tecplot, when reading CGNS files, expect the grid and solution data to be in the same file. Wind-US outputs a solution file separate from the grid input file. To overcome this problem, a small utility program called `link-solution` has been written and distributed with SmaggIce under the `utils` subdirectory. The program creates ADF links within the CGNS grid file that point to the solution nodes inside the solution file (`http://www.grc.nasa.gov/WWW/cgns/adf/subs_structure.html#Link`). After using this program to link the grid and solution files, the grid file appears to the post-processors as a single file containing both grid and solution. The post-processors then work as expected.

When working with CGNS files, you may want to use the `ADFViewer` program, a free viewer for CGNS files (`http://www.grc.nasa.gov/WWW/cgns/cgnstools/adfviewer/`). It includes a utility to export CGNS files as PLOT3D files (`http://www.grc.nasa.gov/WWW/cgns/cgnstools/utilities/export.html`). Downloads of various CGNS utilities are available from `http://www.cgns.org/Utilities.html`.

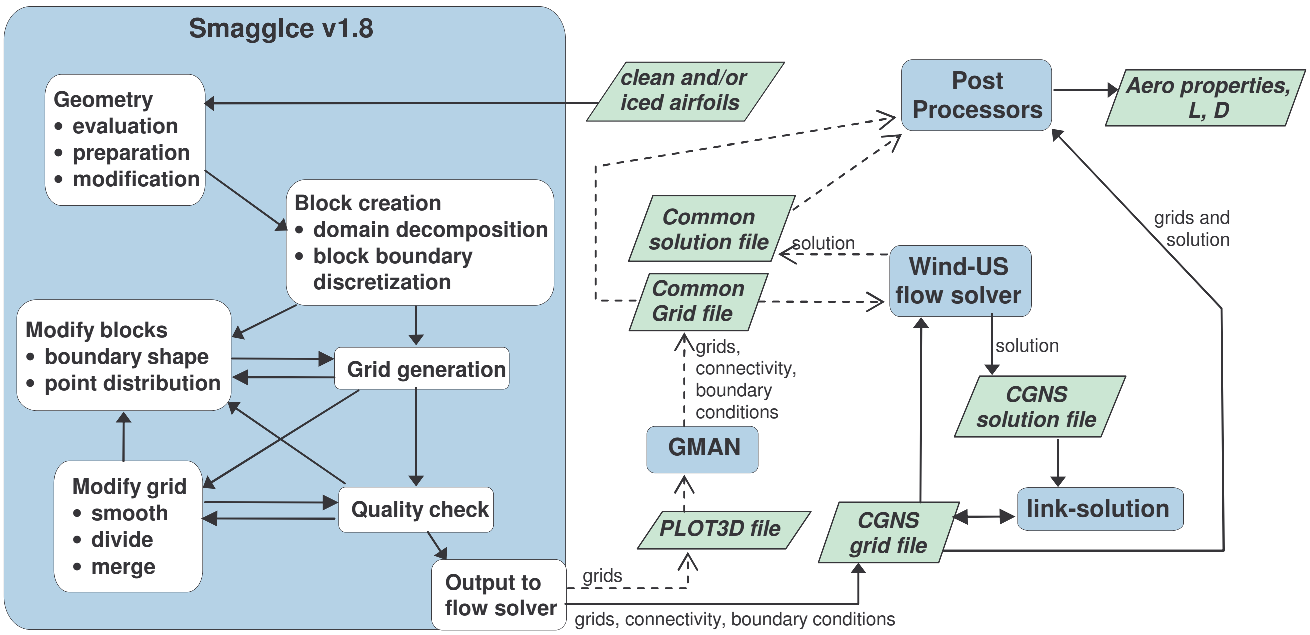


Figure 4. Workflow within SmaggIce and linking with Wind-US flow solver.

User Interface

You will interact with SmaggIce through a Graphical User Interface (GUI) and/or scripts.

GUI (Graphical User Interface)

The GUI includes menus, windows, buttons, text fields, and mouse controls. The GUI is designed to provide "directed control" or "guided use". This means that you will be prevented, through the desensitization or "graying out" of widgets (menus, buttons, sliders, etc.), from selecting conflicting functions or functions that are invalid in certain situations. Error checking is performed at all levels, starting at the GUI, whenever the user enters parameters. Any errors such as out-of-range data or invalid values are immediately reported so that the user can correct them.

The SmaggIce main window is shown below in Figure 5. There are six main parts:

- *Main menu* across the top of the window. Most functions in SmaggIce are requested through menu selections. Many menu selections will bring up secondary windows with which you will interact by entering additional information. Other menu selections will perform a function immediately.
- *Information* area in the upper left corner, in which detailed information about the specific function being performed may be displayed. For example, while making ice measurements, the coordinates of points are displayed here. As another example, when a subcurve is being selected, information on the subcurve endpoints is displayed here.
- *Current Object* information area in the middle of the left column gives information about the object which is currently selected and capable of being modified. This area displays information such as the type of object, its name, and a graphics diagram identifying the object type. For example, when an element is the current object, the

number of points in the element is displayed, along with an indication of whether it is an open element or closed element. When a block is the current object, the number of points on each edge of the block is displayed.

- *Graphics Window Info* area in the lower left corner gives information about the graphics mode and instructions for using the mouse while in that mode. These modes are entered by selecting certain functions in SmaggIce. While in these modes, mouse button presses and movements can be used for interactions in the Graphics Window.
- *Icon buttons* across the top, just below the main menu allow for quick access to commonly-used functions.
- *Graphics Window*, which is the largest area of the display. This is the area in which the geometry is displayed and mouse interactions with the data are performed.

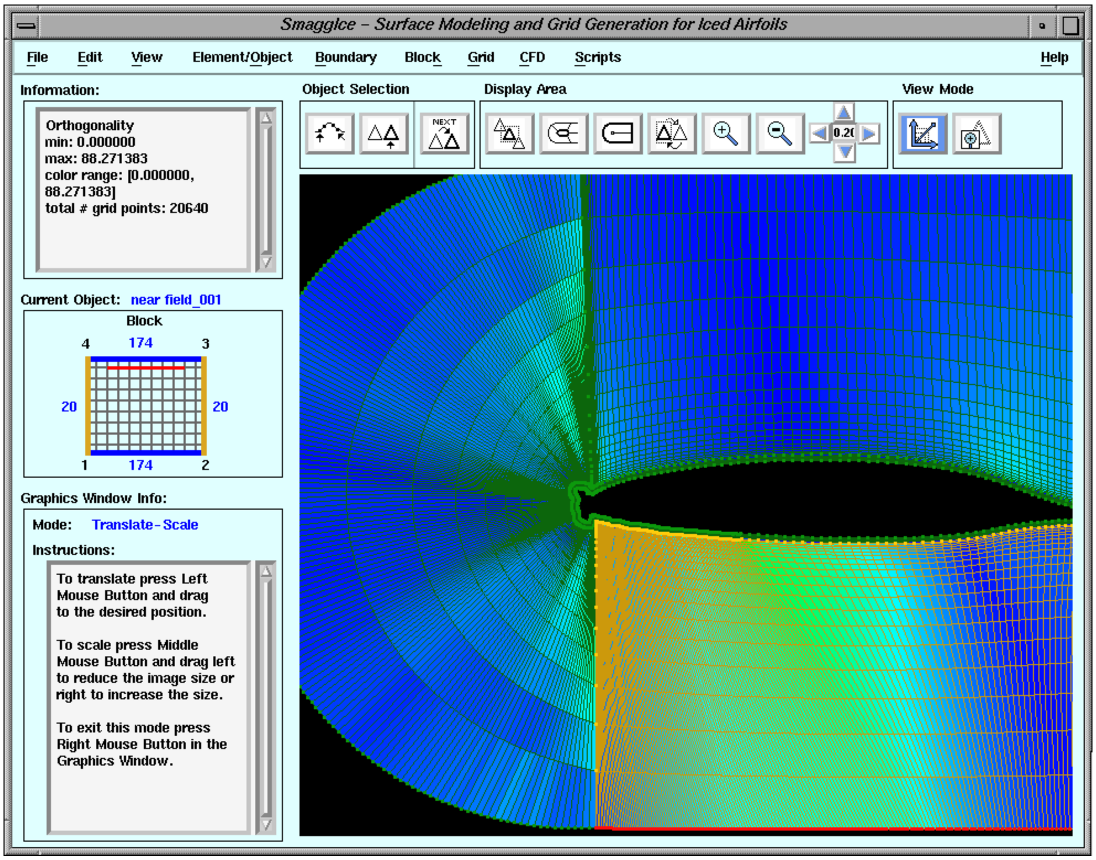


Figure 5. SmaggIce main window

Using Scripts

In addition to using the GUI to manipulate the data in SmaggIce, you can also write and run scripts. SmaggIce lets you run scripts written in the Python programming language (`http://www.python.org`). Python has been extended to include function calls that modify the data in SmaggIce. Sample scripts have been provided in the `scripts` subdirectory under the SmaggIce installation directory. These may be copied and edited as desired using a text editor. Scripts may also be created by recording the actions you perform in the GUI to a script file.

Even if you do not explicitly turn on recording, SmaggIce always logs your actions to files located in the `$HOME/smaggice_logs` directory. Each time you start up SmaggIce, a new log file is created in that directory. These files are named according to the date and time when your session with SmaggIce began. An example of one of these log file names is `Fri_Mar_11_11:30:31_2005.log`. Since these files are constantly being created, you might want to clean out this log directory every so often. Although the **Save State** feature of SmaggIce also lets you start up where you left off, you could also run one of these log files to do the same thing.

You can run a script in SmaggIce by selecting **Scripts→Run Script File** from the main menu. On UNIX systems, you may also run a script when SmaggIce is started, by including it as a parameter to the `runsmagg` command.

The scripting capability in SmaggIce has many benefits:

- Recording your actions in the GUI to a script file allows you to remember the steps you used in a session with SmaggIce.
- Recording scripts is a great way to learn how to write scripts.
- Often, there are a series of steps that you repeat over and over again. These steps can be put into a script and then run at the appropriate time. This is faster and less error prone than using the GUI.
- Script files are a great way to share with others the processing steps you used in SmaggIce.
- Script files can be used for parametric studies. A script can define a basic set of instructions to create a set of grids. By changing a parameter in a script file, you can try the same basic set of instructions with a new parameter.

Notes on Recorded Scripts

Undo/redo in scripts does not work. There is no "undo" or "redo" command in a script. If recording is on and you select **Undo** or **Redo**, what gets recorded is a comment that an undo or redo occurred. Therefore, if you want to run a script that was created by recording, and you did an undo or redo during recording, you need to edit the script file and manually "undo" and "redo" by deleting or duplicating commands in the script file.

Running a script from within another script does not work. There is no "SmgRunScriptFile" command available from within a script. If recording is on and you do **Scripts→Run Script File**, what gets recorded is a comment that a script was run. Therefore, if you want to run a script that was created by recording, and you ran a secondary script during recording, you need to edit the script file and manually include the secondary script file in the one you are editing.

Online Help

SmaggIce online help uses your web browser to display help files, which are stored in HTML format. If you have trouble displaying SmaggIce on-line help on a UNIX workstation, make sure a web browser is available on your workstation, and make sure the environment variable `WEBBROWSER` is set correctly in the file `bin/runsmagg` in the `smaggice` directory. If you cannot display SmaggIce online help on a MS-Windows PC, make sure Internet Explorer is available and that the environment variable `WEBBROWSER` is set correctly in the file `smaggrun.bat` in the `smaggice` directory. You may also view the help files independently of SmaggIce by pointing your browser to the file `help/index.html` in the `smaggice` directory.

Tutorial

This section will step you through some general tasks in SmaggIce, and then present several scenarios: view modification, boundary modification, measuring ice shapes, adding artificial ice, generating a simple grid, and generating a grid on an iced airfoil.

Note: All the geometry files used in this tutorial are available from the `geometry` subdirectory under the SmaggIce installation directory.

General tasks

This section will walk you through some general tasks that you will need to be able to do whenever you use SmaggIce.

Read geometry

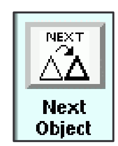
You will read in a three-element airfoil which is stored in a SmaggIce element file. Select from the main menu: **File→Open**.

In the *File Open* window,

- Select **SmaggIce Element** as the *File Type*.
- Read the file: `3elemIced.elt` from the `geometry` directory. The window will close automatically after the file is read in.

Select current object

Objects in SmaggIce can be either elements (such as airfoils or ice shapes) or blocks (which define the flow domain and may contain grids). Most operations in SmaggIce are performed on the current object. You may select the current object using one of these methods:



- To set the next available object as the current object, select from the main menu: **Element/Object→Next Object**, or use the hot key `<Ctrl>N`, or click on the **Next Object** icon at the top of the main window. Do this several times to change the current object.
- To set the previous object in the list of available objects as the current object, select from the main menu: **Element/Object→Previous Object** or use the hot key `<Ctrl>P`.
- To select the current object from the list of available objects, select from the main menu: **Element/Object→Select Object by Name**. In the window that is displayed, click on the object you want to select.
- To select the current object with the mouse, first select from the main menu: **Element/Object→Select Object**, or click on the **Select Object** icon at the top of the main window. You can now position the mouse over an object in the graphics window and click the left mouse button to select it as the current object. Instructions for doing this are displayed in the *Instructions* area in the lower left corner of the main window. Remember to exit this selection mode by pressing the right mouse button when you are finished selecting the object.

Notice that the selected object is displayed in red, and the non-selected objects are displayed in green. Also notice the information that is displayed in the *Current Object Info* area on the left portion of the main window. The name of the object is displayed, as well as the number of points in the element, and an indication of the type of element (open or closed).

Get on-line help

On-line help is available and displayed through your web browser.

- To request general help from the main window, select from the main menu, **Help→Online Help**
- As you start using other SmaggIce windows, you will see that specific help may be accessed by pressing the **Help** button at the bottom of each of those windows.

Clear all objects

At times you will want to clear all graphics and data, and return SmaggIce to its initial state.

- From the main menu, select: **Edit→Clear All Objects**.

Undo

If you make a mistake in SmaggIce, or just want to go back to an earlier state, you can use the Undo capability.

- From the main menu, select: **Edit->Undo Clear All Objects**.
- This is a multi-step undo, so you may perform undo several times. Select: **File->Undo Select Object**.
- The number of undo levels that are available to you is limited, but the limits can be adjusted. From the main menu, select: **Edit->Preferences**. Select *Undo* in the **Category** list, and set the number of undo levels and/or the amount of memory that will be reserved for saving undo information. If you want to save these settings and have them apply during other SmaggIce sessions, press the **Save** button to save them to a file. If you only want them to take effect for this session, press the **OK** button.
- Redo is also available. From the main menu, select: **Edit->Redo Select Object**.

Exit

To exit SmaggIce,

- From the main menu, select: **File→Exit**.
- You will be asked to verify your choice.
- Press **Yes** to exit.

View modification

This section will explain how to change the view you have of the geometry. You will be using the nine widgets across the top of the screen to do this (Figure 6).

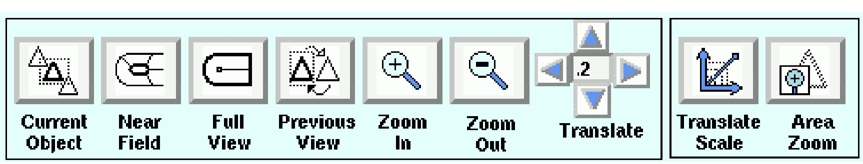


Figure 6. View modification widgets from SmaggIce main window.

Startup

- Start SmaggIce (or select **Edit→Clear All Objects**, if already running a session).
- Read the SmaggIce element file: `3elemIced.elt` from the `geometry` directory.

Graphics Mode transformations

The two icon buttons on the top right of the graphics window (**Translate/Scale**, and **Area Zoom**) set the Graphics Window mode so that mouse movements in the graphics window will change the view. When you select one of these, you will notice that the mouse cursor changes to indicate that you are in a special view manipulation mode.

- Press the **Translate/Scale** icon (notice that it turns blue, to indicate that the Translate/Scale mode is active), then follow the instructions in the lower left corner of the main screen to move the geometry around and enlarge or reduce its size.
- Press **Area Zoom**. This allows you to draw a box around an area that will be enlarged to fill the screen.
- To exit any of these modes, click on the icon again (or click the right mouse button while in the graphics window). The icon will turn gray to indicate that the mode is not active, and the mouse cursor will return to its previous shape.

Immediate transformations

Pressing one of the six icon buttons at the top middle of the graphics window (**Current Object, Near Field, Full View, Previous View, Zoom In, Zoom Out**, and **Translate** arrows) immediately changes your view of the geometry.

- **Current Object** sets the view to include the entire current object.
- **Near Field** sets the view to include all the near field blocks (if those have been created).
- **Full View** resets the view to include all of the geometry.
- **Previous View** sets the view to the one just before the most recent view change.
- **Zoom In** or **Zoom Out** scales the geometry up or down incrementally.
- The **Translate** arrows move the geometry up, down, left, or right by the specified window percentage. You may change the percentage used in translating by typing a new value in the center box of this widget.

Boundary modification

Many SmaggIce functions are performed on subcurves of objects. This section will show you how to select and modify subcurves.

Startup

- Start SmaggIce.
- Read the SmaggIce element file: `NACA23015_2ice_scaled.elt` from the `geometry` directory.
- Notice that there are three objects: the full clean airfoil and two ice shapes. The clean airfoil is a closed element, meaning that its first and last points are at the same location (the trailing edge). The two ice shapes are open, meaning that the first and last points of the elements are not at the same location. Make sure that the clean airfoil is selected. Your geometry should look like that shown in Figure 7.

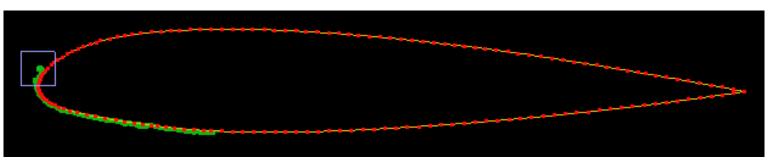


Figure 7. Geometry file: NACA23015_2ice_scaled.elt; clean airfoil selected.

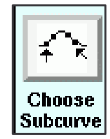
Displaying Anomalies

You will notice that a section of the ice is highlighted with a purple square. This indicates a section of the ice that has a crease in it (where two adjacent line segments make an angle of less than 60 degrees). If there were twists in the geometry, you would see those highlighted in magenta. You can zoom in on the highlighted area to see the actual crease in the geometry. For now, we will turn off display of creases.

- From the main menu select **View→Anomalies→Edge Creases** to deselect that choice.

Select subcurve

Initially, the selected subcurve for an element is the entire curve. You will usually want to select only part of the curve to work with.



- From the main menu, select: **Boundary→Choose Subcurve**, or click on the Choose Subcurve icon at the top of the main window.
- The cursor will change to indicate that you are in a special mode for selecting points. The instructions in the lower left corner of the screen will tell you how to select the endpoints of the subcurve. As you select the endpoints of the subcurve, notice how the information in the upper left corner of the screen changes. Also note that the points on the subcurve are displayed in red, while the points not on the subcurve are displayed in orange.
- Select the upper surface of the airfoil as the current subcurve. This will be between point indices 1 and 83. You may find it easier to select specific points by using the mouse to click near the point, then using the arrow keys to set the exact point. The left and right arrow keys can be used to change the subcurve endpoint to a point next to the most recently selected point.
- To exit the Choose Subcurve mode, click on the Choose Subcurve icon, or press the right mouse button while in the graphics window.

Hyperbolic tangent redistribution

This allows you to redistribute the points in the selected subcurve. This tool will let you concentrate the distribution of points near the leading and/or trailing edge(s).

- From the main menu, select: **Boundary→Tanh Redistribution**.
- Notice that two sets of points are displayed: the current points as red dots and the modified temporary points as blue crosses.
- You may change the amount of stretch and how the stretching is distributed between the two endpoints by using the sliders in this window. You can also change a value by typing a number in the appropriate text entry area and pressing `<Enter>`. Set the *Stretch* value to `100` and the *End Pt 1/End Pt 2* value to `50`.
- If you would like to replace the current points with the modified points, press **Apply**. To close the window without making the changes permanent, press **Close**. Pressing **OK** will apply the changes and close the window all at once. Your geometry should now look like that in Figure 8.

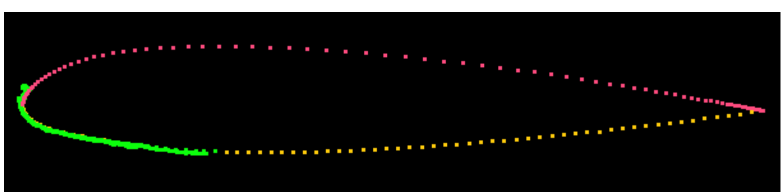


Figure 8. Redistributed points on upper surface of airfoil.

Discretize subcurve

This allows you to change the number of points in the subcurve and how they are distributed.

- Select `element_001` as the current object, and zoom in on the area at the leading edge of the airfoil (Figure 9). Select indices 1 and 56 as the endpoints of the subcurve by using the **Boundary→Choose Subcurve** menu selection.
- From the main menu, select: **Boundary→Discretize Subcurve**. If this menu selection is grayed out, it may be that you are still in the Choose Subcurve mode. (Follow the instructions in the lower left corner of the main screen, and click the right mouse button to exit that mode.)
- Again, the current points are displayed as red dots while the modified points are displayed as blue crosses.

- To change the **Uniform** parameter, drag the slider. To evenly distribute the points, set this to the maximum value of `10`.
- Change the number of points in the subcurve to `100`, by typing that value in the text field and pressing `<Enter>`.

Press **Apply** to make the changes permanent.

- You may want to concentrate the distribution of points near areas of high curvature. Set the **Uniform** parameter back to `0`, then type a value of `20` in the **Curvature** text field and press `<Enter>`.
- Make the changes permanent, or discard the changes the same way you did in the *Tanh Redistribution* window. Figure 9 illustrates these changes from rediscretization.

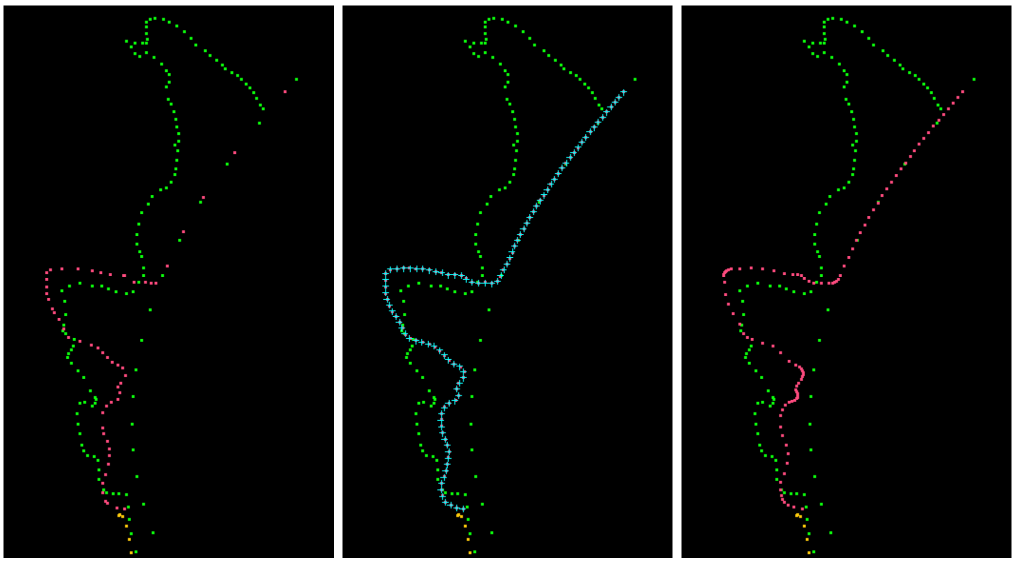


Figure 9. Left to right: original distribution; uniformly discretized to 100 points; discretized by curvature.

Change subcurve using free form method

This allows you to smooth the subcurve and to modify the subcurve by moving control points that define its shape.

- Make sure the view is zoomed into the area at the front of the airfoil (Figure 10).
- Select `element_002` as the current object and select indices 1 and 112 as the endpoints of the subcurve. (Remember to exit Choose Subcurve mode when you are done.)
- From the main menu, select: **Boundary→Change Free Form Subcurve**
- The **check boxes** in this window allow you to display the smoothed (modified) points and the control points, as well as the lines connecting those points. Try turning these on and off.
- Increase the number of **Smoothed** points to `150` by typing that value in the text box and pressing `<Enter>`.

Decrease the number of **Control** points to `40` by typing that value in the text box and pressing `<Enter>`. This has the effect of smoothing the subcurve.

- You may also modify the curve by moving the control points. Press the **Move FF Control Pts** button. You are now in the Move FF Control Points mode. Follow the instructions to move control points. To exit this mode, press the right mouse button.
- Make changes permanent, or discard the changes the same way you did in the *Tanh Redistribution* window. Figure 10 shows what these changes would look like if you made them permanent.
- If you make the changes permanent, turn highlighting of creases back on (**View→Anomalies→Edge Creases**), and you will see that the section of the ice horn that had the crease is no longer highlighted, since smoothing the subcurve removed the crease.

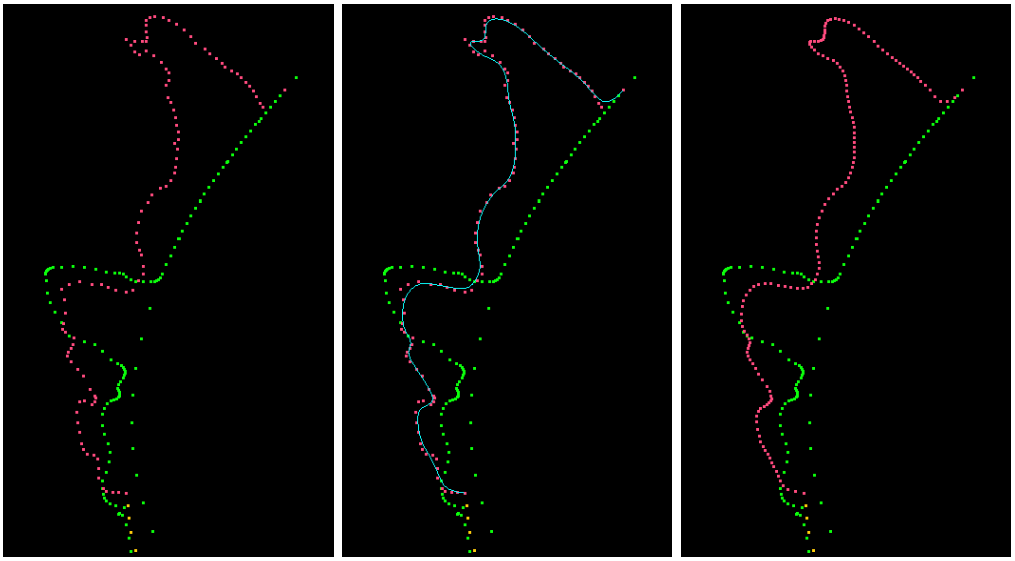


Figure 10. Left to right: original points; curve smoothed using 40 control points (control points not shown); new curve with 150 points.

Save modified geometry

After making these changes to the geometry, you may want to save the geometry to a new file.

- From the main menu, select: **File→Save As**
- Under the *Save* option menu, select **All Elements (SmaggIce)**. Type the name of the new file in the *Selection* text box and click **Save**.
- You may want select **Edit→Clear All Objects** and then read in this new file to verify that the changes were saved.

Measuring Ice Shapes

This section will step you through the process of making measurements of ice shapes for ice shape characterization.

Startup

- Start SmaggIce.
- Read the SmaggIce element file: `NACA23015_3ice_scaled.elt`. This file contains 4 closed elements: a clean airfoil and three iced airfoils.
- Zoom in to the area at the leading edge of the airfoil.
- Turn off highlighting of edge creases (**View→Anomalies→Edge Creases**).
- Select `element_001` to be the current object.

Display point coordinates

You can display the coordinates of points in the current object. When you select a point, its index and its (x,y) coordinates are displayed in the *Information* area of the main screen.

- From the main menu, select: **View→Display Point Coordinates**
- Follow the instructions in the lower left corner of the screen to select points and display their coordinates in the *Information* area. These coordinates are for immediate reference only. The following steps will show you how to keep a record of point locations, as well as other types of measurement information.
- To exit Display Point Coordinates mode, press the right mouse button.

Select reference airfoil

For some measurements, such as *(X-Xle)/C*, a clean airfoil must be identified as a reference, since the chord length *C* is determined from the clean airfoil. The chord length can also be used to normalize measurements such as length. In order to choose these calculations, you must first identify the clean airfoil so that the chord length can be determined.

- Set the current object to the clean airfoil, named `element`.
- From the main menu, select: **Element/Object→Set Reference Airfoil**.
- The reference airfoil is highlighted and also shows two additional points marked with purple crosses: the leading edge point, and the center of the leading edge circle. While making measurements, you may select these points as you would any others. You may turn highlighting of the reference airfoil on and off from the main menu: **View→Highlight Reference Airfoil**.

Make measurements

- From the main menu, select: **Element/Object→Measure Ice Shapes…**
- Measurements will be recorded in the *Probe Info Table*.
- Comments may be entered at any time into the *Probe Info Table*. Type "`Measurement Tutorial`" in the comment text area, and press the `<Enter>` key or the **Enter Comment** button.

Location

Make a location measurement:

- Select **Location (X,Y)** from the *Probe Measurement* option menu. Follow the instructions in the *Instruction* area of the main window to select a point. Now select a different point. Notice how the probe information is displayed in the *Information* area of the main window. However, nothing is entered into the *Probe Info Table* until you press the right mouse button. Now enter the location into the table by pressing the right mouse button.

Length

Make a measurement of normalized length:

- Change the *Probe Measurement* to **Length**.
- You are currently able to choose points from all objects. Now change the *Point Selection Method* by selecting **Current Object** from that option menu. This will limit the points which may be selected to those on the current object only.
- Click on the *Normalize* check box to measure normalized length. This will normalize all measurements by dividing them by the chord length, which in this case is 8.0002.
- Type the string "`LE circ rad`" into the *Label* text field, indicating that this will be a measurement of the radius of the circle inscribed in the leading edge of the airfoil. This label will be entered into the *Probe Info Table* when you enter the measurement.
- Follow the instructions to select two endpoints: one being the leading edge of the airfoil, the other being the center of the leading edge circle (both marked in purple crosses). Enter the measurement into the table. Note that if the reference airfoil is not highlighted, you will not be able to select these two points.

Glyphs

You will have noticed that after measurements are entered into the *Probe Info Table*, symbols (or glyphs) such as points and lines are overlaid on the geometry to indicate where the measurements were made. You may turn on and off the display of these glyphs by selecting from the main menu: **View→Show→Measurement Glyphs**.

Arc length

Arc length measures the sum of the lengths of the line segments along a boundary between two points on that boundary.

- Set the *Probe Measurement* to **Arc Length**, and turn *Normalize* off.
- Select the two endpoints: one being the leading edge of the reference airfoil, the other being the location where the ice attaches to the airfoil at the upper surface.

- Set the *Label* to "`ice limit`", and enter the measurement into the table.

Angle

To measure an angle, you will need to mark two lines between which the angle will be measured. Normalizing this measurement has no effect.

- Set the *Probe Measurement* to **Angle**.
- Following the instructions in the *Instruction* area, set the endpoints of the first line: the center of the leading edge circle and the leading edge point itself.
- For the second line, set the first endpoint to again be the center of the leading edge circle. For the second endpoint, set the *Point Selection Method* to **All Objects**, and select the tip of the uppermost ice horn.
- Set the *Label* to "`upper horn`", and enter the measurement into the table.

(X-Xle)/C

This measurement calculates the value `(X-Xle)/C`, where `X` is the x-location of the point selected, `Xle` is the x-location of the leading edge, and `C` is the chord length of the reference airfoil.

- Set the *Probe Measurement* to **(X-Xle)/C**.
- Set the *Label* to "`ice tip`", select the point on the tip of the uppermost ice horn, and enter the measurement into the table.

Ice Area

Ice area measures an area between an ice shape and the reference airfoil. This measurement can only be made if there is a reference airfoil and at least one other element.

- Select `element_003` to be the current object.
- Set the *Probe Measurement* to **Ice Area**.
- Set the *Point Selection Method* to **Current Object**, so that only points on the ice shape may be chosen.
- Follow the instructions to select two endpoints on the ice shape. Lines will be drawn from the selected points to the closest point on the reference airfoil. The area bounded by those lines, the reference airfoil, and the iced airfoil will be computed. If *Normalize* is on, that area is divided by the area of the reference airfoil.
- Type the string "`Ice area`" into the *Label* text field and enter the measurement into the table by clicking the right mouse button in the Graphics window.

Figure 11 shows what the measurement glyphs look like when they are displayed. Figure 12 shows the Probe Info Table with the measurements entered.

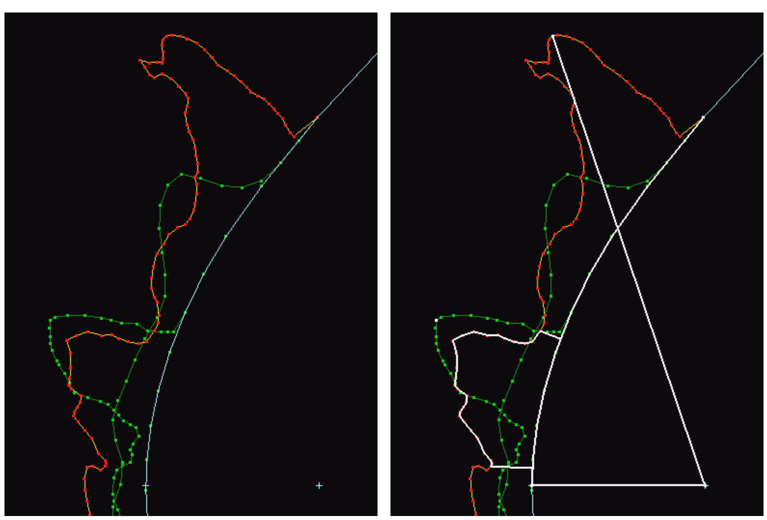
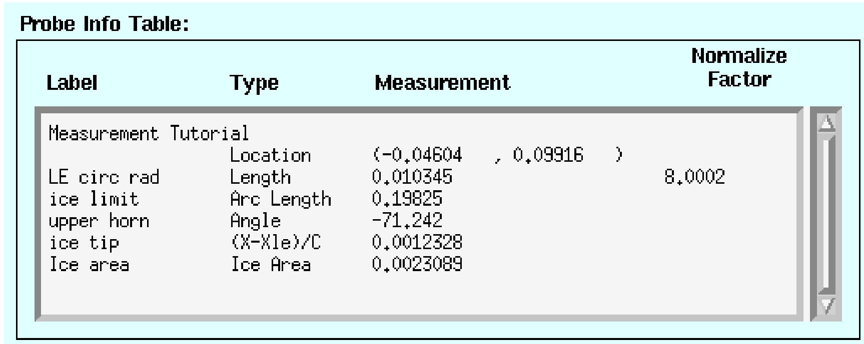


Figure 11. Left: highlighted reference airfoil with markers at leading edge and center of leading edge circle; Right: glyphs marking measurements of location, length, arc length, angle, (X-Xle)/C, and ice area.



Probe Info Table:

Label	Type	Measurement	Normalize Factor
Measurement Tutorial			
	Location	(-0.04604 , 0.09916)	
LE circ rad	Length	0.010345	8.0002
ice limit	Arc Length	0.19825	
upper horn	Angle	-71.242	
ice tip	(X-Xle)/C	0.0012328	
Ice area	Ice Area	0.0023089	

Figure 12. Probe Table entries in the Measure Ice Shapes window.

Save probe information

At this point, you may want to save the probe information to a file.

- In the *Measure Ice Shapes* window, press the **Save…** button. This will bring up a window which will allow you to save the *Probe Info Table* to a text file.
- You may also save a screen image of the geometry with the lines and points of your measurements displayed. First make sure glyph display is turned on by selecting **View→Show→Measurement Glyphs** from the main menu, then select **File→Screen Save**. You may choose to save the image as a TIFF, GIF, or PPM image file.
- To clear the *Probe Info Table* and all associated glyphs, press the **Clear All** button in the *Measure Ice Shapes* window.

Add Artificial Ice

This section will show you how to add computer-generated artificial ice shapes to an airfoil.

Startup

- Start SmaggIce (or select **Edit→Clear All Objects**, if already running a session).
- Read the SmaggIce element file: NACA23015`_clean.elt`.
- Locations and sizes of artificial ice are specified in chord units. Set the reference airfoil first so that the chord length can be calculated. From the main menu select: **Element/Object→Set Reference Airfoil**.

Add Spoiler

Artificial ice is added to the current object. Since there was only one element read in, it is already set as the current object.

- From the main menu, select: **Element/Object→Add Artificial Ice**.
- Select the **Rectangle** button.
- Values are already filled in for all the parameters which define the shape and the location. You can change any of these values and press `<Enter>` to see a preview. Type a value of `.02` for the *X-location* and press `<Enter>`.

At this point you will see a small rectangle on the upper surface of the airfoil near the leading edge.

- Change the *Height* to `.08`, *Angle* to `10`, and the *number of points in the Y-direction* to `8`. Notice that as you type values in the text fields, no changes are displayed until you press `<Enter>`. Now press `<Enter>`. You will see a preview of the points on the spoiler displayed as blue crosses.
- To make these changes permanent, press the **Apply** button.

Add Train of Shapes

A set of shapes may be replicated along the surface of the airfoil. We will create a train of semi-circles along the lower surface of the airfoil.

- Press the **Semi-circle** button.
- Select **Lower Surface** for the *Location*.
- Enter the following values:
  X-location: `.1`
  # Shapes: `10`
  Distance Between: `.01`
  Radius: `.02`
- Press `<Enter>` to see a preview.
- Click the **OK** button to make the changes permanent and close the window.

Your geometry should look like that illustrated in Figure 13.

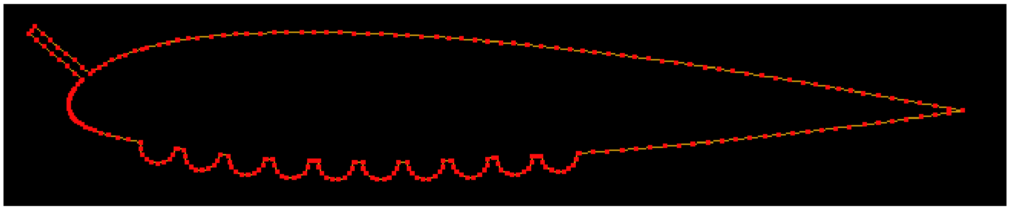


Figure 13. Clean airfoil with spoiler on upper surface near leading edge and train of semi-circles along lower surface.

Move Element

This section will show you how to translate the current element and rotate it about a hinge point.

Startup

- Start SmaggIce (or select **Edit→Clear All Objects**, if already running a session).
- Read the SmaggIce element file: `3elemIced.elt` from the `geometry` directory.
- Notice that there are three closed objects: the slat, the main airfoil, and the flap. Make sure that the slat is selected.

Move Element

- From the main menu select: **Element/Object→Move Element** (or press `<Ctrl>V`).
- The hinge point is drawn as a small pink dot. For the rotation of the slat, use the default hinge point location of `(0,0)`. Note that for other geometries, you may need to specify a different hinge point location. For instance, for the rotation of the flap, you might use `(0.87, 0.00146)`.
- Change the *Rotation* parameter: drag the slider to the left to `-20` or enter the value in the adjacent text field and press `<Enter>`. Notice the slat is rotated around the hinge point location `(0,0)`. Translate the object by entering a value of `.05` for *X-translation* and `.05` for *Y-translation*. Press `<Enter>` to see a preview of the new location, shown in blue. Your geometry should look like that illustrated in Figure 14.

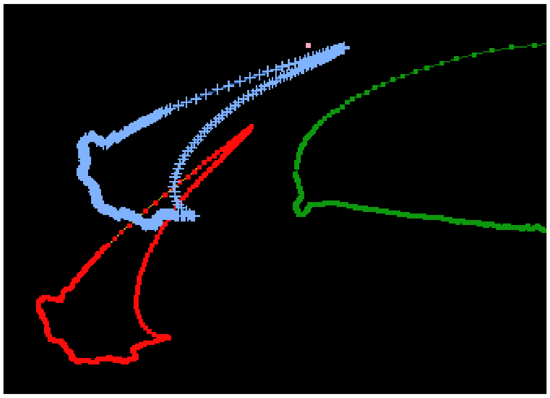


Figure 14. Preview of slat rotated and translated.

- Press the **Apply** button to make the changes permanent. The window will stay open to allow you to make further changes. When you are finished press the **Close** button. If you want to make the changes permanent and close the window in one step, press the **OK** button.

Generate Simple Grid

This section will show you how to create a simple grid on a clean airfoil.

Startup

- Start SmaggIce.
- Read the SmaggIce element file: NACA23015`_clean.elt` from the `geometry` directory.
- The chord length is needed to create the wake, so set the reference airfoil (which calculates the chord length). From the main menu select: **Element/Object→Set Reference Airfoil**.

Create Wake

Creating the wake is the first step in domain decomposition.

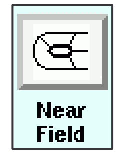
- From the main menu select: **Block→Wake Definition.**
- Enter the following values:
  Number of Points: `30`
  Length: `10.0`
  Angle: `0.`
- Press `<Enter>` to see a preview.
- Click the **OK** button to make the changes permanent and close the window.

Create the Viscous Sublayer Block

- From the main menu select: **Block→Viscous Sublayer.**
- Enter the following values:
  JMAX: `20`
  Thickness: `.001`
  Minimum Grid Spacing: `.000002`
- Press `<Enter>` to see a preview.
- Click the **OK** button to make the changes permanent and close the window.

Create the Near Field Blocks

- We will create a near field with two radial cuts extending from the trailing edge, resulting in a near field made up of three blocks. Using radial cuts allows for better control of point distribution on the outer edge of the near field blocks.
- From the main menu select: **Block→Near Field Decomposition.**
- From the pull-down menu at the top of the *Near Field Decomposition* window, select **Radial Cut**.
- Enter the following values:
  Radius: .7
  JMAX: 30
  Number of Radial Cuts: 2
- Press <Enter> to see a preview.
- Click the **OK** button to make the changes permanent and close the window.
- From the main menu select **View→Display Area→Near Field**, or click on the Near Field icon at the top of the main window. This will change the view to include the near field area.



Redistribute Points of Near Field Block

- The points along the outer edge of the near field block that surrounds the airfoil could be distributed better. Right now they are clustered too much at the leading edge.
- Make the block near field_001 the current object. How to select the current object is described in the General Tasks section above.
- Select the outer edge of this block as the current subcurve. There are two ways to do this. One is to select **Block→Next Edge** or <Ctrl>E multiple times until the desired edge is highlighted. The other way is to use **Boundary→Choose Subcurve** to select all the points on that edge. You may find this somewhat tricky, because if you click on a corner point, there are two edges it can choose from and the wrong one might be selected. One easy way to avoid this problem is to use the mouse to select a point on that edge that is close to the corner, and then use the arrow keys on the keyboard to extend the subcurve to the corner point. When you are done, the graphic window should look like Figure 15.

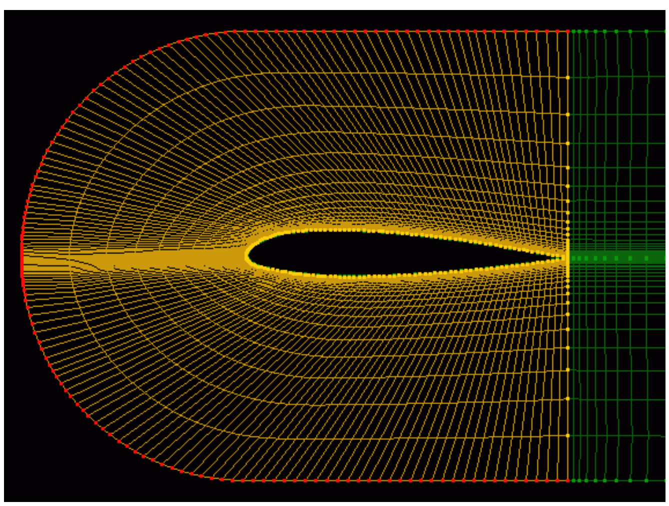


Figure 15. Block "near field_001" with JMAX (outer) edge selected.

- Redistribute the points on this edge so that the spacing matches that of the adjacent blocks. From the main menu, select **Boundary→Redistribution to Match Cell Spacing**. In the window that is displayed, select both **End Pt1** and **End Pt2**. This will show a preview.
- Click the **OK** button to make the changes permanent and close the window.
- The grid was removed when you modified the boundary points, so now you have to regenerate it. From the main menu, select **Block→Generate Grid→Current Block**.

Merge Grids

Now we will merge all the near field blocks into a single block.

- From the main menu, select **Grid→Merge Grids**. The grids that are able to be merged with the current grid will be highlighted with a gray overlay. Click on the block that is above the wake with the left mouse button. You will see a preview of the merged grid. Press the middle mouse button to make the merge permanent and exit this mode.
- Repeat this process to merge the block below the wake with the current block. You now have a single near field block.

Create Outerblock

- From the main menu, select **Block→Outerblock Definition**.
- Enter the following values:
  Outerblock: `1`
  Near Field: `1`
  Outer Radius: `15.0`
  IMAX: `150`
  JMAX: `20`
- Press `<Enter>` to see a preview.
- Click the **OK** button to make the changes permanent and close the window.

The grids should look like those shown in Figure 16.

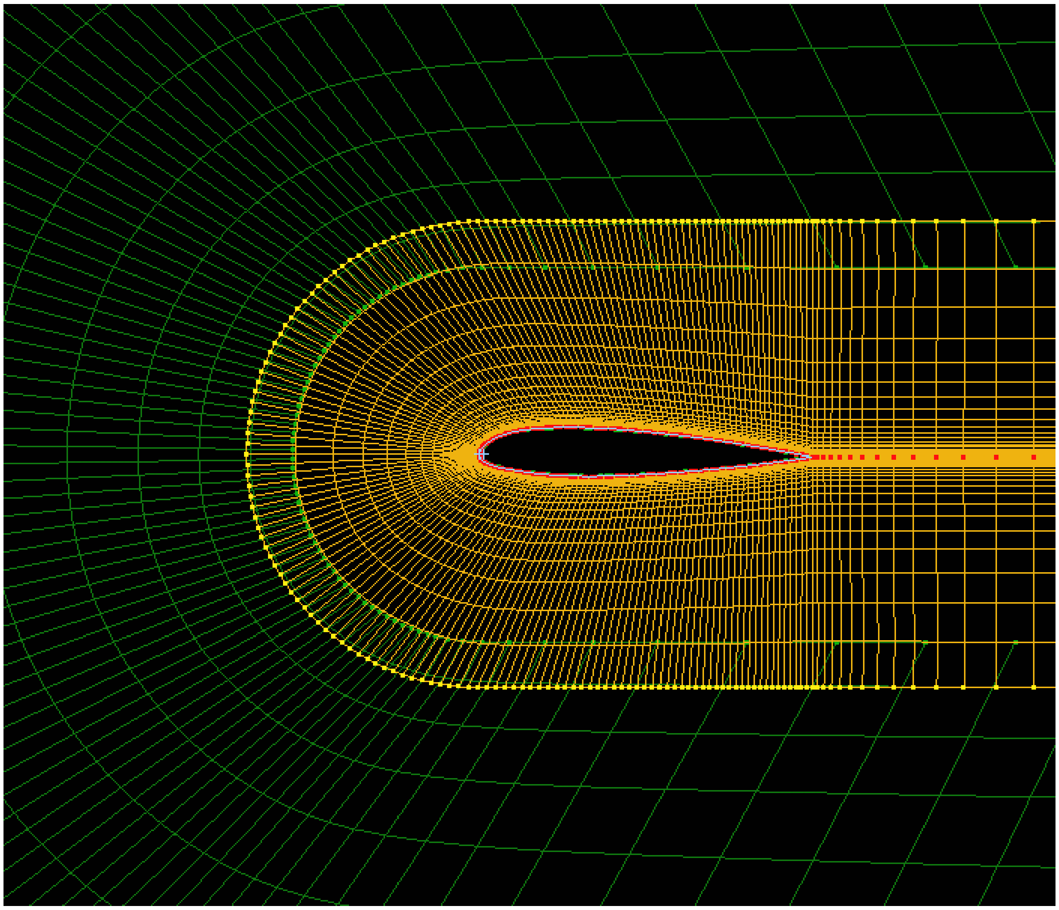


Figure 16. Single near field block with redistributed points and overlapping outerblock.

Save the Grids

We will save the grids to a PLOT3D file. This file is 2D, formatted, multi-zoned, non-Iblanked.

- From the main menu, select **File→Save As**.
- Select *All Grids (Plot3D)* from the **Save** pull-down menu.
- Enter a valid file name in the **Selection** box, and press the **Save** button.

Generate Grid for Glaze Iced Airfoil

This section will show you how to create a grid on an airfoil with a somewhat complex ice shape.

Startup

- Start SmaggIce.
- Read the SmaggIce element file: `623glaze_GA_partial.elt` from the `geometry` directory. This contains a clean airfoil and a partial ice shape.

Initial Geometry Preparation

- The clean airfoil is not closed (i.e., the points at the trailing edge do not meet). We will extend the trailing edge to close this element. From the main menu select **Element/Object→ Extend Trailing Edge→Linear**.
- Set the clean airfoil as the reference airfoil, since it will be used in the next step. From the main menu select **Element/Object→Set Reference Airfoil**.
- We want to adjust the chord length, changing it from the value that was calculated for us automatically. From the main menu select **Element/Object→Set Chord Length/LE**. Enter `1.0` as the *Chord Length* and press **OK**.
- Select the next object (the ice shape) as the current object. **Element/Object→Next Object**. This is only a partial ice shape and must be extended before it can be blocked and gridded. From the main menu select **Element/Object→Extend Ice using Ref. Airfoil**.

Redistribute Points on Iced Airfoil

We would like to distribute the points on the entire iced airfoil surface more uniformly. Right now, the entire surface is selected as the current subcurve, so we do not have to reselect any points.

- From the main menu select **Boundary→Rediscretize Subcurve**.
- Enter the following values:
  Uniform: `6`
  Curvature: `0.0`
  Num. of Points: `584`
- Press `<Enter>` to see a preview, then press the **OK** button.

Next we will apply stretching to the point distributions on the upper surface of the airfoil.

- First select the points along the upper surface of the airfoil. You will select the subcurve with endpoint indices of 1 and 197. From the main menu select **Object/Element→Choose Subcurve**.
- *Endpoint 1* is already selected at index 1. So we only need to reselect *Endpoint 2*. Place the pointer near the location where the ice horn meets the upper surface of the airfoil, and click with the middle mouse button (to pick *Endpoint 2*). The information window will display the endpoints that you have chosen. If you didn't click on point with index 197 exactly, just use the left and right arrow keys on the keyboard to decrease or increase the index of the selected point. When you have the correct subcurve selected, press the right mouse button.
- To stretch the points using the Hyperbolic Tangent Redistribution method, from the main menu select **Boundary→Tanh Redistribution**.

- Enter the following values:
  Stretch: `21`
  End Pt 1 / End Pt 2: `100`
- Press `<Enter>` to see a preview, then press the **OK** button.

Finally, we will apply stretching to the point distributions on the lower surface of the airfoil.

- Select the points along the right-most ⅔ of the lower surface of the airfoil. You will select the subcurve with endpoint indices of 444 and 584. From the main menu select **Object/Element→Choose Subcurve**.
- Use the left mouse button to select *Endpoint 1* about ⅓ of the way back toward the trailing edge. Remember to check the index displayed in the *Information* area, and use the left and right arrow keys on the keyboard to adjust the point selection. Use the right mouse button to select *Endpoint 2*. You will not be able to choose the trailing edge point on the lower surface, because when you click on that point, the point on the trailing edge of the upper surface (which is in the same location) will be selected. Rather, select a point on the lower surface near the trailing edge, and then use the arrow keys to move the selection out to the trailing edge. Press the right mouse button when you are done selecting the subcurve.
- From the main menu select **Boundary→Tanh Redistribution**.
- Enter the following values:
  Stretch: `15`
  End Pt 1 / End Pt 2: `0`
- Press `<Enter>` to see a preview, then press the **OK** button.

Create Wake

- From the main menu select: **Block→Wake Definition.**
- Enter the following values:
  Number of Points: `40`
  Length: `15.0`
  Angle: `0.`
- Press `<Enter>` to see a preview.
- Click the **OK** button to make the changes permanent and close the window.

Create the Viscous Sublayer Block

- From the main menu select: **Block→Viscous Sublayer.**
- Enter the following values:
  JMAX: `20`
  Thickness: `.006`
  Minimum Grid Spacing: `.00001`
- Press `<Enter>` to see a preview.
- Click the **Apply** button to make the changes permanent, but do not close the window. The graphics window should contain an image similar to Figure 17.

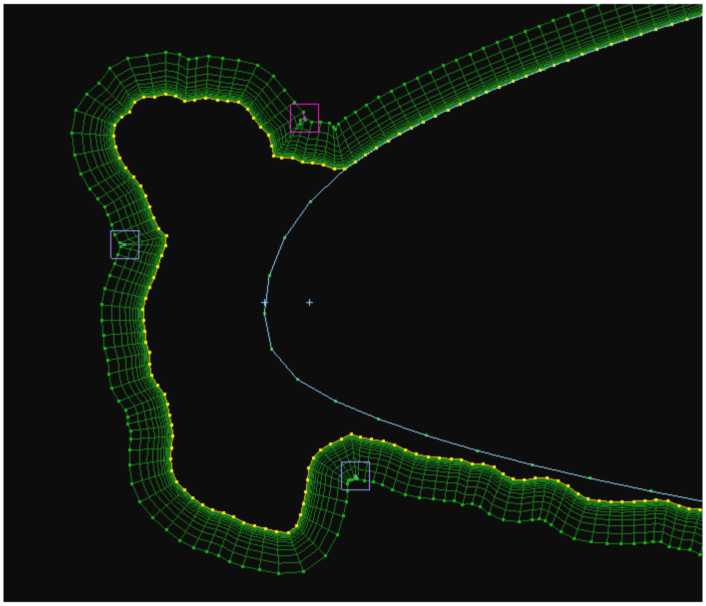


Figure 17. Twists and creases on the JMAX boundary of the viscous sublayer are highlighted.

You will see sections of the viscous sublayer highlighted where there are twists or creases. These twists and creases show up in concave areas because of the thickness of the viscous sublayer. These may be removed by decreasing the thickness of the viscous sublayer, but you may want to try another option. SmaggIce can remove these automatically by selectively smoothing the ice shape in problem areas.

- Click on the **Remove Twists** check box in the *Viscous Sublayer* window. You can zoom in on the highlighted areas to see how the ice surface and the grid have been modified. Click the **OK** button to make the changes permanent and close the window.

Modify the Viscous Sublayer Block

We still need to do a little more work on concave areas of the outer boundary of the viscous sublayer. We will select small subcurves and smooth them slightly.

- Select the viscous sublayer block as the current object (**Element/Object→Next Object**).
- Select the subcurve (**Boundary→Choose Subcurve**) on the JMAX (outer) edge of the viscous sublayer with endpoint indices 233 and 248. This is on the upper surface above where the upper ice horn meets the clean airfoil. Don't forget to exit Choose Subcurve mode.
- From the main menu select **Boundary→Change Free Form Subcurve**. You will see that the grid is removed, in anticipation of the block boundary changing.
- Set the **Num. Control Pts** to `3`. Reducing the number of control points has the effect of smoothing the curve. Press `<Enter>` and then **OK**. The graphics window should show an image similar to Figure 18.

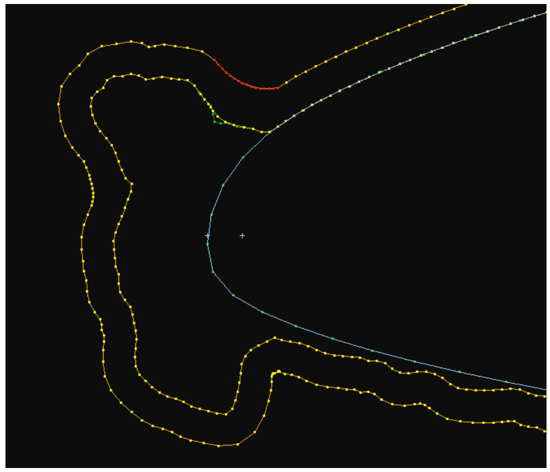


Figure 18. Subcurve on JMAX boundary of viscous sublayer after smoothing.

- Select the subcurve on the JMAX (outer) edge of the viscous sublayer with endpoint indices 311 and 323. This is on the lower surface below where the lower ice horn meets the clean airfoil.
- From the main menu select **Boundary→Change Free Form Subcurve**.
- Set the **Num. Control Pts** to `3` to smooth the curve. Press `<Enter>` and then **OK**.

Regenerate the grid using the new boundaries:

- From the main menu select **Block→Generate Grid→Current Block**.

Create the Near Field Blocks

We will create a near field with six radial cuts, resulting in a near field made up of seven blocks.

- From the main menu select: **Block→Near Field Decomposition**.
- From the pull-down menu at the top of the *Near Field Decomposition* window, select **Radial Cut**.
- Enter the following values:
  Radius: `.6`
  JMAX: `40`
  Number of Radial Cuts: `6`
- Press `<Enter>` to see a preview.

These radial cuts are not placed in the optimal locations. Let's place and shape them to fit our ice geometry.

- Click the **Move Critical Pts** button in the *Near Field Decomposition* window.

Small pink squares indicating the location of the critical points will appear on each radial cut. These may be dragged to reposition or shape the cut. Moving critical points on the ends of the radial cuts will reposition the cut and draw a straight line between the two endpoints. Moving critical points on the line itself will reshape the cut as a smooth curve whose shape is controlled by the critical points.

- Move the critical points to approximate those shown below in Figure 19. Especially concentrate on the area near the iced airfoil, to make sure the cuts meet the airfoil at an angle close to 90 degrees. See the close-up in Figure 20 for details.
- Press the right mouse button to exit the Move Critical Points mode, and press the OK button to make the changes permanent and close the window.

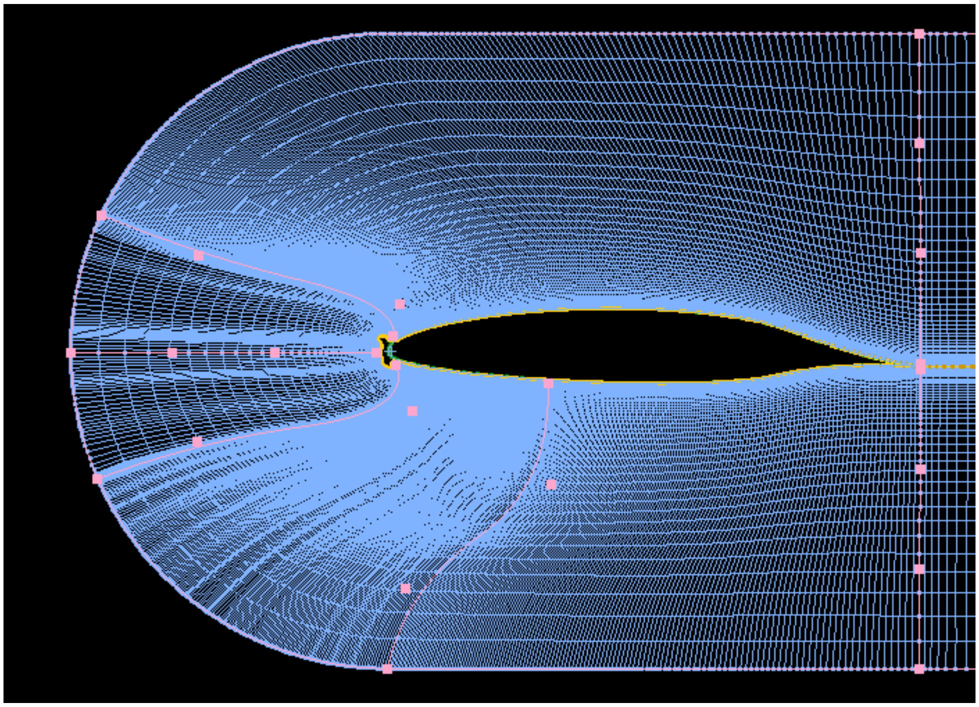


Figure 19. Radial cuts shaped using critical points.

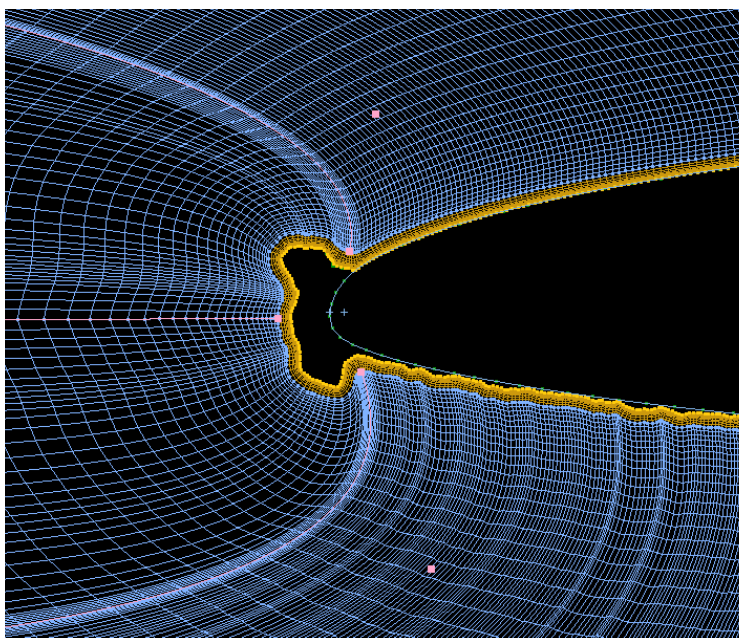


Figure 20. Close up of radial cuts where they meet the JMAX boundary of the viscous sublayer.

Modify the Near Field Blocks

Next we will redistribute points on the JMAX (outer) edges of three of the near field blocks (`near field_002`, `near field_003`, and `near field_004`) to make the point spacing uniform.

- Select the JMAX edge of block `near field_002`. You can do this by first selecting the block as the current object, and then selecting **Block→Next Edge** twice.

- From the main menu select **Boundary→Rediscretize Subcurve**.
- Set the value of *Uniform* to `10` and press the **OK** button.
- Repeat this process for the JMAX edge of blocks `near field_003` and `near field_004`.
- Regenerate the grids for the blocks that do not have them by selecting from the main menu **Block→Generate Grid→Non-Gridded Blocks**.

Now we will make the point spacing change smoothly by setting the point spacing at both ends of the current edge (the JMAX edge of block `near field_004`) to match the point spacing on the adjacent grids.

- From the main menu select **Boundary→Redistribute to Match Cell Spacing**.
- In the window, select both *End Pt1* and *End Pt2* to match.
- Press the **OK** button to make the changes permanent and close the window.
- Regenerate the grid for this block (**Block→Generate Grid→Current Block**.
- Select the previous block (`near field_003`), and repeat that process to match the cell spacing on both ends of the JMAX edge with the adjacent grids. Be sure to regenerate the grid when you are done.

The two near field blocks at the leading edge need to have some smoothing applied.

- Select the entire IMAX edge of `near field_003`. This is the edge that extends directly out from the front of the iced airfoil. See Figure 21.

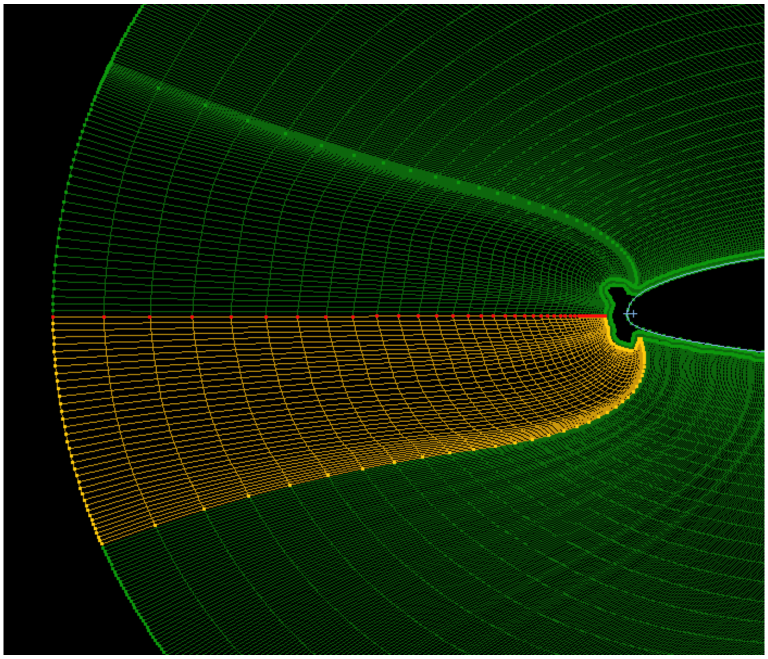


Figure 21. IMAX edge of block "near field_003" selected.

- First we will smooth between the two adjacent blocks. From the main menu select **Grid→Smooth Across Blocks**.
- Enter the following values:
  Number of Cells Current Grid: `8`
  Number of Cells Abutting Grid: `8`
  Number of Iterations: `7`
- Press `<Enter>` to see a preview and the press the **OK** button to make the changes permanent and close the window.
- Now merge the current grid with the adjacent one (`near field_004`) that shares the edge that we just smoothed across. From the main menu, select **Grid→Merge Grids** and select the grid above the current one. Press the middle mouse button to make the merge permanent.

- As the final step, apply smoothing to the newly-merged grid. From the main menu select **Grid→Smooth Current Grid (Elliptic)**.

You might find that the parameters used for generating grids and applying elliptic smoothing do not meet your needs. You may change many of the parameters used in the grid generation and elliptic smoothing process.

- From the main menu select **Grid→Elliptic Parameters**.
- After changing the parameters, press **Apply** or **OK**.
- The next time you regenerate a grid or perform elliptic smoothing, the new parameters will be used.

Examine Grid Quality

Many types of grid quality may be examined. Grid quality can indicate what areas of the grid need further modifications. We will look at orthogonality and cell shape.

- From the main menu select **View→Grid Quality→Orthogonality**. The range of measurements will be displayed in the information window. Grid cells with the highest value of orthogonality will be displayed in red while those with the lowest value will be displayed in blue. Other colors in the rainbow spectrum indicate intermediate values.
- You may find it easier to see the colors of individual cells if you turn off display of the grid lines. From the main menu select **View→Show→Grid** to turn display of the grid off and on. The graphics window should look similar to Figure 22.

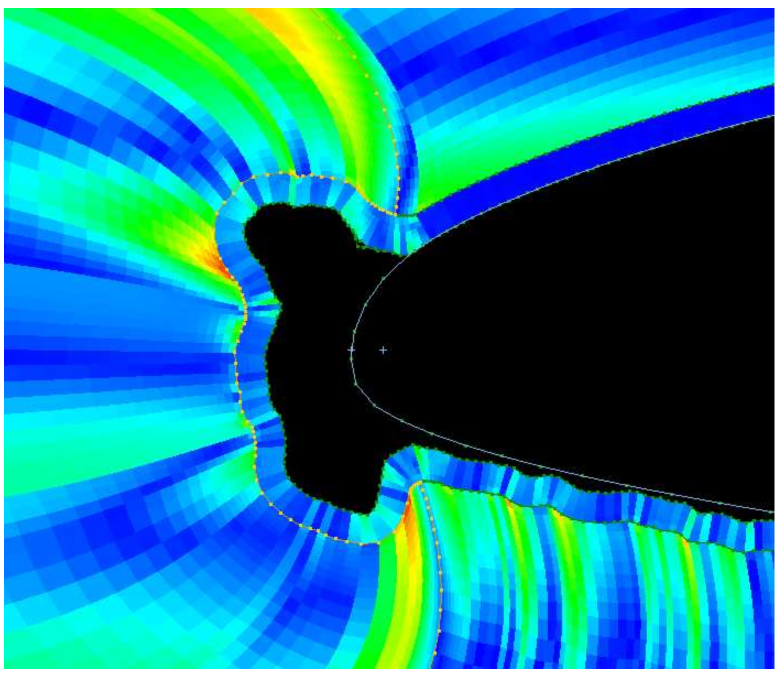


Figure 22. Grid cells colored by orthogonality; grid lines turned off.

One very valuable use of grid quality is to make sure all the grid cells have a valid shape.

- Turn grid display on again (**View→Show→Grid**).
- From the main menu select **View→Grid Quality→Cell Shape**. The information area will display a message indicating how many (if any) invalid cells are in your grids. If there are invalid cells, they will be highlighted in magenta. If you do have invalid cells, you must fix them before and outerblock can be created.

Create Outerblock

- From the main menu, select **Block→Outerblock Definition**.

- Enter the following values:
  Outerblock: `1`
  Near Field: `2`
  Outer Radius: `15.0`
  IMAX: `150`
  JMAX: `20`
- Press `<Enter>` to see a preview.
- Click the **OK** button to make the changes permanent and close the window.

Save Your Work

You can save your work at any time to a State File, and read it back in later to continue working where you left off. Let's save what we have done so far.

- From the main menu, select **File→Save State**.
- Enter a valid file name in the **Selection** box, and press the **Save** button.
- When running SmaggIce at a later time, you can read this state file back in by selecting from the main menu **File→Restore State**.

Save a CGNS File

A CGNS file stores the grids, connectivity between grids, and boundary conditions. It can be used as input into the Wind-US flow solver.

- From the main menu select **File→Save As**.
- Select *All Grids (CGNS)* as the **Save** type.
- Enter a valid file name in the **Selection** box, and press the **Save** button.

At this time, you may also want to create an input file that will control a Wind-US flow solver run.

- From the main menu, select **CFD→Create WIND Input File**.
- This file can be edited by hand to change any necessary parameters before submitting it, along with the CGNS file, to the Wind-US flow solver.

Generate Grid for Rime Iced Airfoil

This section will show you how to create a grid on an airfoil with a rime ice shape.

Startup

- Start SmaggIce.
- Read the SmaggIce element file: `212rime_busJet.elt` from the `geometry` directory. This contains a clean airfoil and an iced ice airfoil.

Initial Geometry Preparation

- The clean airfoil is not closed (i.e., the points at the trailing edge do not meet). We will extend the trailing edge to close this element. From the main menu select **Element/Object→ Extend Trailing Edge→Linear**.
- Set the clean airfoil as the reference airfoil, since it will be used in the next step. From the main menu select **Element/Object→Set Reference Airfoil**.
- Select the next object (the iced airfoil) as the current object. **Element/Object→Next Object**. This element is not closed and must be extended before it can be blocked and gridded. From the main menu select **Element/Object→Extend Trailing Edge→Linear**.

Redistribute Points on Iced Airfoil

We would like to distribute the points on the entire iced airfoil surface more uniformly. Right now, points 2 through 484 are selected as the subcurve. We will not change this.

- From the main menu select **Boundary→Rediscretize Subcurve**.
- Enter the following values:
  Uniform: `7`
  Curvature: `0.0`
  Num. of Points: `483`
- Press `<Enter>` to see a preview, then press the **OK** button.

Create Wake

- From the main menu select: **Block→Wake Definition.**
- Enter the following values:
  Number of Points: `40`
  Length: `15.0`
  Angle: `0.`
- Press `<Enter>` to see a preview.
- Click the **OK** button to make the changes permanent and close the window.

Create the Viscous Sublayer Block

- From the main menu select: **Block→Viscous Sublayer.**
- Enter the following values:
  JMAX: `20`
  Thickness: `.005`
  Minimum Grid Spacing: `.00001`
- Click on the **Remove Twists** check box.
- Click the **OK** button to make the changes permanent and close the window.

Create the Near Field Blocks

We will create a near field with five radial cuts, resulting in a near field made up of seven blocks.

- From the main menu select: **Block→Near Field Decomposition**.
- From the pull-down menu at the top of the *Near Field Decomposition* window, select **Radial Cut**.
- Enter the following values:
  Radius: `.6`
  JMAX: `30`
  Number of Radial Cuts: `5`
- Press `<Enter>` to see a preview.

These radial cuts are not placed in the optimal locations. Let's place and shape them to fit our ice geometry.

- Click the **Move Critical Pts** button in the *Near Field Decomposition* window.
- Move the critical points to approximate those shown below in Figure 23.
- Press the right mouse button to exit the Move Critical Points mode, and press the OK button to make the changes permanent and close the window.

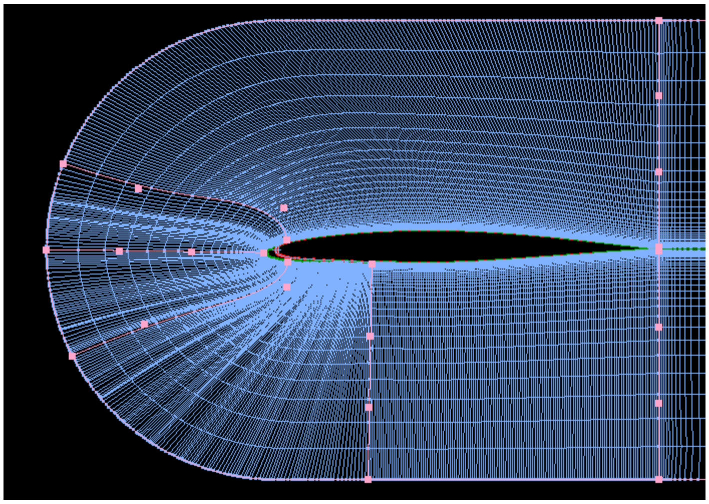


Figure 23. Radial cuts shaped using critical points.

Modify the Near Field Blocks

We will merge the two near field blocks at the leading edge and smooth that merged block.

- From the main menu select: **Element/Object→Select Object By Name**. Click on `near field_003`. This will select the lower block at the leading edge of the airfoil. Close the selection window.
- From the main menu select: **Grid→Merge Grids**. Merge the current block with the block just above it at the leading edge.
- To smooth this merged grid, from the main menu select: **Grid→Smooth Current Grid (Elliptic)**. Do this a second time to smooth even more.

Examine Grid Quality

Make sure all the grid cells have a valid shape.

- From the main menu select **View→Grid Quality→Cell Shape**. The information area will display a message indicating how many (if any) invalid cells are in your grids. If there are invalid cells, they will be highlighted in magenta. If you do have invalid cells, you must fix them before and outerblock can be created.

Create Outerblock

- From the main menu, select **Block→Outerblock Definition**.
- Enter the following values:
  Outerblock: `1`
  Near Field: `2`
  Outer Radius: `15.0`
  IMAX: `150`
  JMAX: `20`
- Press `<Enter>` to see a preview.
- Click the **OK** button to make the changes permanent and close the window. Your grids should look similar to the ones in Figure 24.

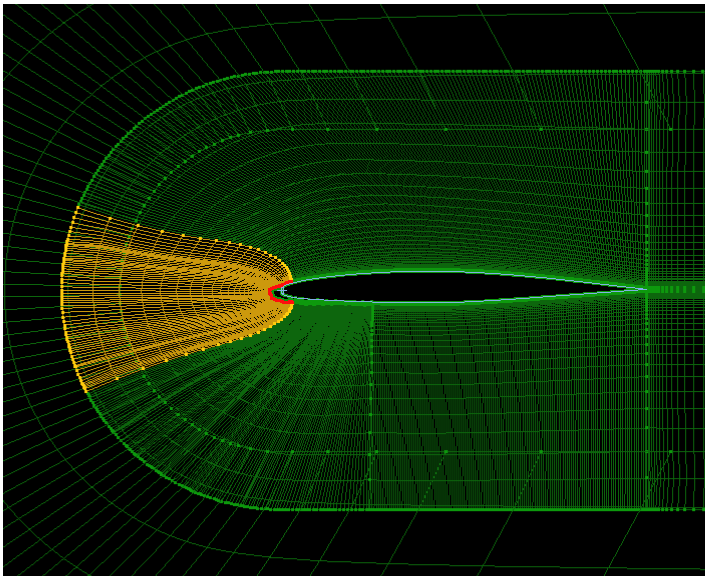


Figure 24. Gridded airfoil with rime ice.

Save a CGNS File

- From the main menu select **File→Save As**.
- Select *All Grids (CGNS)* as the **Save** type.
- Enter a valid file name in the **Selection** box, and press the **Save** button.

At this time, you may also want to create an input file that will control a Wind-US flow solver run.

- From the main menu, select **CFD→Create WIND Input File**.
- This file can be edited by hand to change any necessary parameters before submitting it, along with the CGNS file, to the Wind-US flow solver.